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Model manipulation

MODUS

Model manipulation

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Contents

1   Model manipulation

1.1   Tutorial pre-requisites

- The student must have completed the 'Basic options within the new inspection dialogue' tutorial

1.2   Tutorial objectives

- To familiarise the student with all the tools available to manipulate models for programming aid, customise views and use the model for program execution aids

- To introduce the student to using models as devices and the options available using a model device

2   Introduction

This tutorial explores the methods available to utilise CAD functionality in MODUS to aid part programming and promote efficient operation.

3   Moving using icons and shortcut keys

The following options are available from the model window toolbar for manipulating the view of the model.

NOTE: 'Orbit', 'Pan', 'Zoom' and 'Zoom to Box' remain selected until they are deselected. It is not possible to pick from the model until they have been deselected.

To manipulate the view of the model use mouse and key combinations:

Pan = CTRL + mouse wheel click

Rotate = Mouse wheel click

Zoom = Mouse wheel rotate

Select - Displays a sub-menu with options for selecting geometry entities on the CAD model. It is possible to then hide these or hide the unselected entities. For further information on this feature, refer to 'Selecting, hiding and deleting model entities' later in this tutorial.

Orbit - To orbit the contents of the model window. Click the 'Orbit' button, and then drag the mouse cursor in the model window.

To rotate the model around a particular position on the model, ensure the 'Orbit' button is selected, and then click on the model. A marker is displayed on the model at the location of the click. When the model is dragged, it rotates about the marker.

To change the location of the marker, with the 'Orbit' button selected click on another part of the model.

To cancel the tool, click the 'Orbit' button again to deselect it, or click one of the other buttons on the model window toolbar.

Also, to rotate the contents of the model window, press and hold the middle mouse button, then drag. To do this the 'Orbit' button must NOT be selected. If the centre of rotation is on the CAD model, this is indicated by a marker on the CAD model.

Pan - Allows repositioning of the CAD model in the window using the mouse. Click the 'Pan' button, and then drag the CAD model to reposition it. Or press and hold the 'CTRL' key while pressing and holding the middle mouse button, then dragging.

Zoom - Using the 'Zoom' command (in conjunction with the 'Pan' and 'Orbit' commands) allows the view to be manipulated. This is useful for picking in areas of the CAD model which may be concealed in the current view, or in areas of fine detail. Click the 'Zoom' button and push the mouse forwards while pressing and holding the left mouse button to zoom in. To zoom out, drag the mouse backwards while pressing and holding the left mouse button, or rotate the middle mouse wheel when 'Zoom' is selected.

Zoom to Box - Click the 'Zoom to Box' button and drag the mouse cursor to draw a box over the area of the model to expand. Or press and hold the 'ALT' key while pressing and holding the middle mouse button, then dragging.

Zoom In / Out - Click the 'Zoom In' and 'Zoom Out' buttons to zoom in and out in steps in the window.

Zoom to Extents (Full View) - Expands or reduces the model to fit the window. Or double-click the middle mouse button.

View - Displays the various surface views of the model. The isometric option displays the model so that lines of equal length along each of the three axes look the same length. The Z-axis is vertical, the Y-axis is at 4 o'clock and the X-axis is at 8 o'clock.

Axis Type - Allows selection of the axes to use when displaying the various views and rotating the model. Selecting a different axis type may not immediately change the model in the model window. Instead, it defines the active axis system for the next manipulation of the model.

Model Axes - Select this option to display the various views and to rotate the model with respect to the model (part) axes. Click the 'View Full' button, this expands or reduces the model to fit the model window.

Machine Axes - Select this option to display the various views and to rotate the model with respect to the machine axes. Click the 'View Full' button, this expands or reduces the image of the machine and model to fit the model window.

Datum Axes - Select this option to display the various views and to rotate the model with respect to the datum axes. Click the 'View Full' button, this expands or reduces the model to fit the model window with respect to the datum axes.

Cutting Axes - Select this option to display the various views and to rotate the model with respect to the cutting axes. Click the 'View Full' button, this expands or reduces the model to fit the model window with respect to the cutting axes. This option is useful when cutting planes are active, for example when picking the features in the model window for valve seat inspection. It is selected by default when cutting planes are used.

Wireframe - Displays the model with transparent faces. This allows features not visible on the real part to be seen through obscuring faces.

Rendered (shaded) - Displays the model with opaque, coloured faces. The faces are shaded with reference to a simple light source.

Hidden Line - Displays the model with opaque but uncoloured faces, so that features through obscuring faces cannot be viewed.

Transparent - Displays the model with transparent but coloured faces. This allows features not visible on the real part to be seen through obscuring faces but in a more solid form than the wireframe view. This can be useful when inspecting an internal cylinder, for example, to check that any intersecting holes are avoided by a helical scan path.

To adjust the degree of transparency, press and hold the 'CTRL' key while rotating the middle mouse wheel. The model window must be selected to do this.

Cutting plane - Allows creation of a cutaway view of the model. MODUS creates a plane on the model, in the plane of a picked feature, and allows movement of the plane through the model, removing the part of the model in front of the plane. This gives a view of the inside of the model, as though part of the model had been cut away.

SHIFT + mouse wheel rotate = Move cutting plane

SHIFT + double mouse wheel click = Invert cutting plane

CTRL + double mouse wheel click = Rotate cutting plane through 90°

Print Feature (Graphic Report) - With graphical reporting enabled, prints the displayed feature to a .html file and to the printer specified using the 'DMIS Output Print Set-Up' option from the 'File' menu.

Increase Graphic Report Magnification - With graphical reporting enabled, click to magnify the feature deviations. The magnification value is shown in the model window.

Decrease Graphic Report Magnification - With graphical reporting enabled, click to decrease the magnification of the feature deviations. The magnification value is shown in the model window.

Graphic Report Options - Displays a sub-menu to switch on and off the display of the graphical reporting items drawn in the model window.

4   'Save View as Picture' option

Save the current model window view as a .jpeg image, to obtain a snapshot of the current view to refer to later, email or include in reports. Right-click on the CAD model window and select 'Save View As Picture' from the pop-up menu.

This displays the 'Save As' dialogue box, then browse for a location to save the picture in. MODUS provides a default name for the picture using the model filename (without the extension) displayed in the title bar. If no model is open, MODUS uses CAD_Model as the name. Overtype the name to change it if necessary.

Ensure that the view in the model window is as required before taking the snapshot. For example, to zoom in, do this in the model window before saving the view. This can provide a clearer image than taking the snapshot and then zooming in on the .jpeg image in a screen capture utility, as the image may degrade when doing this.

5   3D mouse

MODUS is also compatible with a space / 3D mouse. This is a second 'mouse' which is purely there to manipulate the CAD model without having to select any of the icons or shortcut keys (the CAD model manipulation methods shown below can be changed using the mouse software).
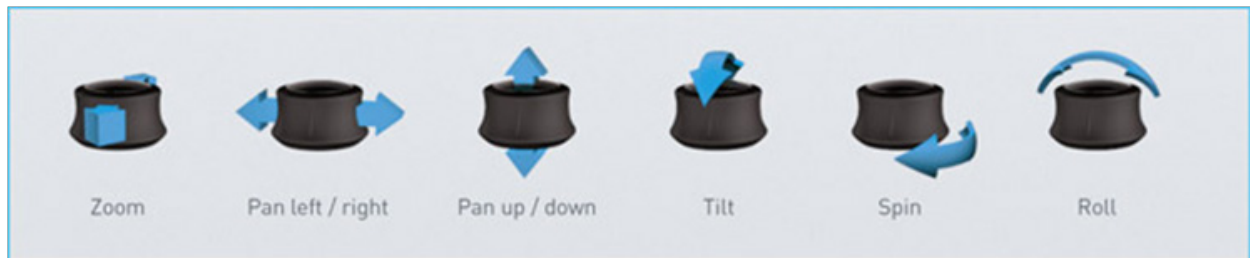
6   Selecting, hiding and deleting model entities

The 'Select' option allows selection of geometry entities on the CAD model. Once selected, these can be hidden, unhidden, and deleted. When a face is selected, the edges of the face are also selected. Free curves can be selected individually. The 'Select' option is intended for use with the model in rendered (shaded) view, or wireframe view, for ease of picking on the model. It is available from the toolbar in the model window, or from the right-click pop-up menu:
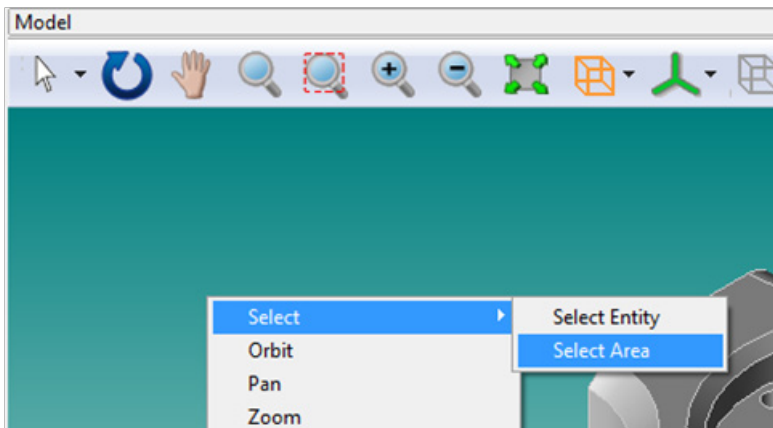
Two options are available from the 'Select' menu:

Select Entity - Select this option to pick individual face or edge entities on the CAD model. Click the entity to be selected on the model. To add further entities to the selection, use CTRL + click or SHIFT + click. To deselect a specific entity, CTRL + click on it.

Select Area - Select this option to drag the mouse cursor to draw a box over the area of the model that are to be selected. All entities that are wholly or partially within the box are selected.

When entities on the model have been selected, use the other toolbar buttons in the model window to adjust the view of the model. The selected entities remain selected. Click the 'Select Entity' option or the 'Select Area' option again, to continue selecting entities, using CTRL + click or SHIFT + click to add to the selection as required. To cancel all of the selections, click in the model window outside the model.

4.1   Hiding and unhiding the selected entities

Once the entities have been selected, use the 'Edit Model' options available from the right-click pop-up menu to determine which entities are 'hidden' and 'unhidden':

Clear Selection - Deselects all of the selected entities on the model. Also deselect all entities by clicking in the model window.

Hide Selected - Hides the selected entities on the model, i.e. makes them invisible.

Hide Unselected - Hides the unselected entities and displays only the selected entities.

Toggle Hidden - Toggles the display of entities between the selected entities only and the unselected entities only.

Unhide - Displays all the model entities including any that were hidden.

These options allow definition of which entities are visible for later activities.

7   Surface measurement - adding points to the model

The surface measurement commands allows creation of points on the CAD model (or in the model window), to define the measurement path when inspecting certain feature types. There are various ways of creating the points, and a combination of the different ways can be used:

*   The 'Grid' options allows generation of a grid over the model, so touch points can be built or point features on the model from the grid nodes (i.e. the intersections of the grid lines).

*   The 'Curve' options allows a curve on the model to be picked, so touch points can be built, point features and edge points on the model from the curve divisions.

*   Points from a .pts file can be imported, to add to the model

7.1   Adding points from a .pts file

When inspecting multi-point features, add points from a .pts file to the model window. Select 'PTS' from the 'Surface Measurement' commands. In the 'Open' dialogue box, browse for the .pts file. The points are added to the 'Teach Path' view.

NOTE: The units must be set correctly in the part program to match the units for the .pts file.

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pts - An ASCII file listing the co-ordinates, or the co-ordinates and vector, of points. This file can include the feature name. The name can optionally be added to the end of the line containing the co-ordinates (or the co-ordinates and vector). If the name includes spaces, it is truncated to the first space.

Example:

2.0 (the software version number)

0 23.4 67.8 98.1 (the 0 at the start of this line indicates that only x, y, z co-ordinates follow)

1 23 657 922.52 0 0 1 (the 1 at the start of this line indicates that co-ordinates and vector follow)

0 23.4 8.90 98.1 POINT1 (this line shows co-ordinates and point name)

1 23.56 657 922.52 0 1 0 VECTORPOINT1 (this line shows co-ordinates, vector and point name).

8   Model explorer dialogue box
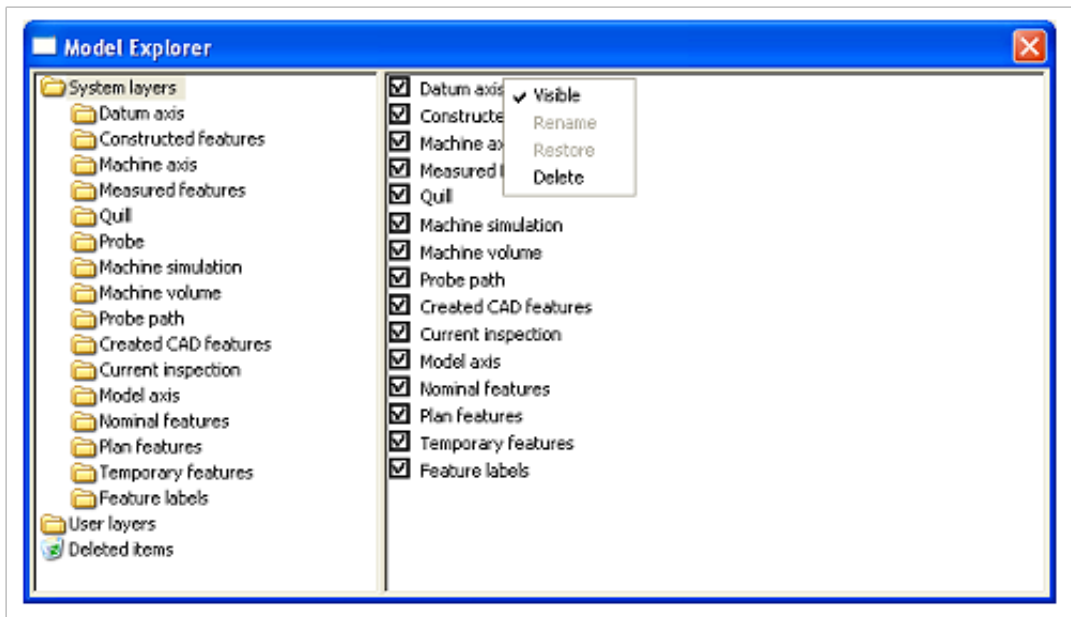
8.1   Purpose

Use the 'Model Explorer' dialogue box to organise the layers of information in the model window. Switch layers on and off to give a clear view of an area or detail of the CAD model, and delete layers or their contents. All the layers assigned to the current CAD model are listed. If the box associated with a layer is checked, then that layer is visible on the CAD model. If it is not checked, then the layer is invisible.

NOTE: When adding slices to the CAD model using the 'Add Slice' dialogue box or the 'Add Circular Slice' dialogue box, slices are only added to visible layers. If the model or any layers are not displayed in the model window, slices are not added to them.

Use the right-click menu on a highlighted layer to access the options available to that layer. Also, right-click on the entities within a layer.

Create new layers using the 'Organise Geometry' dialogue box.

Restore previously deleted items as and when required.

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